Available online at www.sciencedirect.com
ScienceDirect
Procedia Engineering 96 (2014) 400 – 409
Procedia Engineering
www.elsevier.com/locate/procedia

Modelling of Mechanical and Mechatronic Systems MMaMS 2014

Chosen numerical algorithms for interval finite element analysis

Milan Sága[a]*, Milan Vaško[b], Peter Pecháč[c]

Department of applied mechanics, Faculty of Mechanical Engineering, University of Žilina, Univerzitná 1 Žilina 01026, Slovakia
[a]*milan.saga@fstroj.uniza.sk*, [b]*milan.vasko@fstroj.uniza.sk*, [c]*peter.pechac@fstroj.uniza.sk*

Abstract

Our paper presents chosen computational algorithms for solution of finite element models with structural uncertainties. An application of the chosen approaches will be presented – the first one, a simple combination of only inf-values or only sup-values; the second one presents full combination of all inf-sup values; the third one uses the optimizing process as a tool for finding out an inf-sup solution and last one is the Monte Carlo method as a comparison tool.

Keywords: uncertain mechanical parameters, Monte Carlo method, interval arithmetic, optimization algorithm

Nomenclature

$\mathbf{K}$	stiffness matrix
$\underline{\mathbf{K}}, \overline{\mathbf{K}}$	infimum and supremum of the stiffness matrix
$\boldsymbol{u}$	displacement vector
$\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}$	infimum and supremum vectors of the displacement vector
$\boldsymbol{f}$	load vector
$\underline{\boldsymbol{f}}, \overline{\boldsymbol{f}}$	infimum and supremum of the load vector
$\mathbf{M}$	mass matrix
$\underline{\mathbf{M}}, \overline{\mathbf{M}}$	infimum and supremum of the mass matrix
$\boldsymbol{v}$	eigenvector

* Corresponding author. Tel.: +421 41 513 2950; fax: +421 44 565 2940.
E-mail address: milan.saga@fstroj.uniza.sk

doi:10.1016/j.proeng.2014.12.109

$\underline{\boldsymbol{v}_j}, \overline{\boldsymbol{v}_j}$	infimum and supremum of the *j*-th eigenvector
$\boldsymbol{x}$	vector of uncertain structural parameters
$\underline{\boldsymbol{x}}, \overline{\boldsymbol{x}}$	infimum and supremum of the vector of uncertain structural parameters
Greek symbols	
σ	mechanical stress
$\underline{\sigma_j}, \overline{\sigma_j}$	infimum and supremum stresses in elements (resp. nodes)
λ	eigenvalue
$\underline{\lambda_j}, \overline{\lambda_j}$	infimum and supremum of the *j*-th eigenvalue

1. Uncertainty of finite element models

Finite element method (FEM) nowadays belongs to the most significant tools influencing modern design. Although during practical realization, problems related to the lack of input data (boundary conditions, material properties, load) can be often encountered, and these problems can significantly influence the results. Ignoring inaccuracies, which arise from the nature of used method, the relevance of input data is the dominating factor of the FEM's usage effectiveness. This caused a recent increase in quantity of scientific theses and studies dealing with the issue of vagueness of parameters in finite-element models. Apart from the more familiar probabilistic approaches and Monte Carlo analyses are today also applied a bit nontraditional, but also attractive approaches based on fuzzy sets or interval numbers [1, 2]. These mathematical tools are related to the development of the possibility theory as a certain alternative to the probability theory, although the difference between probability and possibility has to be distinguished [3].

Uncertainties in parameters of a computational model can be found in the numerical tool itself (numerical solver, rounding errors), or in the physical nature of the problem. Other possible sources of uncertainties and errors in finite-element analyses are uncertainties of model, discretization error, uncertainties of the input parameters, etc. [4].

Description and definition of these uncertainties and errors is presented in the table 1.

Tab. 1 Uncertainties and errors in finite-element analyses

Type	Description and definition
uncertainty of model	related to the suitability of mathematical model describing a real physical problem
discretization error	related to the conversion from mathematical model to numerical model
uncertainty of parameters	originates from inaccurate knowledge or definition of the input parameters
rounding error	numerical error, defined by natural accuracy of computational arithmetic (method)

1.1. Mathematical and physical uncertainty of models

The first step in the FEM analysis is the choice of an appropriate mathematical model representing the physical nature of a problem. The real problem is although often simplified and idealized and later described by an appropriate mathematical formulation (theory of elasticity, theory of thin plates, equations of thermal conductivity etc.). By the term of uncertainties of model are defined uncertainties, which define how well does the mathematical model represent the behavior of a real physical model [5]. Typical uncertainties in FEM model can be:

- idealization of boundary conditions,
- use of 2D model instead of 3D,
- use of linear model, resp. linearization of nonlinear problems,
- use of time independent model.

1.2. Uncertainties in parameters

Uncertainties in parameters arise, when the accurate values of data, which are needed for an analysis are not available. This type of uncertainties is labeled as parametric or data uncertainties. In finite element analyses can the uncertainties in parameters exist in geometrical, material or input data. The origin of parametric uncertainties can be also caused by lack of information (epistemic uncertainty), or incidental variability of parameters (random uncertainties), or by combination of both. The epistemic uncertainty is bound to the model of a system or a process and its parameters, which arise from limitations in the input data or from inaccurate model definition. It is possible to decrease the impact of this uncertainty by gaining more accurate knowledge about the function of the system (process) itself, or by acquiring more accurate data about an uncertain parameter, which can be incorporated into the mathematical model additionally. The analysis of finite-element models with parametrical uncertainties, algorithms and chosen applications will be in the center of this paper.

1.3. Discretization error

Commonly used mathematical model is represented by finite-element discretization. That includes a certain choice during meshing and element creation. The solution achieved through the use of finite-element method is principally just an approximation of an accurate solution of a mathematical model, and their mutual deviation is defined as the discretization error [6].

The finite-element solution is always affected by number of factors, like for example the number of elements used for mesh generation, type, resp. formulation of the used finite elements, the rules used for integration and many other details in formulation and properties applied during the creation of a computational model.

1.4. Rounding error

The suitability of application of finite-element analysis is limited by finite precision of computational arithmetic. If the arithmetic operations are executed on numbers with floating decimal mark, the accurate result will not be represented as a number with floating decimal mark. The accurate result will be rounded to the closest number with fixed decimal mark and this loss of information is called the rounding error.

To avoid possible serious rounding errors in the process of analysis, the FEA usually requires a high number of significant digits. Thus the use of interval arithmetic is a more appropriate approach. As it has been stated before, the interval arithmetic can limit the rounding error more accurately.

2. Review of chosen computational algorithms for static and modal interval FE analysis

According to the character of the uncertainty we can define a structural uncertainty (geometrical and material parameters) and an uncertainty in load (external forces, etc.). The uncertain structural parameters are usually written into a vector $\boldsymbol{x} = \left[\underline{\boldsymbol{x}}, \overline{\boldsymbol{x}}\right]$ and the interval static (time independent) FE analysis may be formulated as follows

$$\boldsymbol{K}(\boldsymbol{x}) \cdot \boldsymbol{u} = \boldsymbol{f}(\boldsymbol{x}), or \left[\underline{\boldsymbol{K}}, \overline{\boldsymbol{K}}\right] \cdot \left[\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}\right] = \left[\underline{\boldsymbol{f}}, \overline{\boldsymbol{f}}\right], \tag{1}$$

Considering a dynamic conservative system, the interval modal and spectral matrices can be obtained by solving:

$$\left[\boldsymbol{K}(\boldsymbol{x}) - \lambda_j \cdot \boldsymbol{M}(\boldsymbol{x})\right] \cdot \boldsymbol{v}_j = \boldsymbol{0}\, , or \, \left(\left[\underline{\boldsymbol{K}}, \overline{\boldsymbol{K}}\right] - \left[\underline{\lambda_j}, \overline{\lambda_j}\right] \cdot \left[\underline{\boldsymbol{M}}, \overline{\boldsymbol{M}}\right]\right) \cdot \left[\underline{\boldsymbol{v}_j}, \overline{\boldsymbol{v}_j}\right] = \boldsymbol{0}, \tag{2}$$

It is well-known that the classical interval arithmetic application for FE analysis is very limited. Its "overestimating" grows with the problem's size (dimension of system matrices) and really has no physical foundation. Therefore, it is effective to apply special numerical methods [7, 8].

Next we will concentrate on a presentation of the developed numeric methods, which were implemented into interval finite element analyses. We will analyze computational algorithms programed by authors [9, 10].

2.1. Monte Carlo algorithm (MC)

The Monte Carlo algorithm is based on search of the resulting inf-sup solution by generation of a large set of solutions and an estimation of result for extreme values. The advantage of this algorithm is that it will find inf-sup solution, which is located in the acceptable region and not only in the boundary points.

2.1.1. Static analysis:

1st step: Generation of a random matrix (uniformly distributed)

$\mathbf{X}_{\mathrm{MC}} = [\boldsymbol{x}_1, \ldots \boldsymbol{x}_{\mathrm{m}}], (\mathrm{m} \approx 5000 \div 100000),$

2nd step: Calculation of the displacement matrix:

$\mathbf{U}_{\mathrm{MC}} = [\mathbf{K}(\boldsymbol{x}_1)^{-1} \cdot \boldsymbol{f}(\boldsymbol{x}_1), \ldots, \mathbf{K}(\boldsymbol{x}_{\mathrm{m}})^{-1} \cdot \boldsymbol{f}(\boldsymbol{x}_{\mathrm{m}})],$

3rd step: Calculation of inf-sup displacements $[\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}]$:

- Finding the infimum:
  - for i-th node $\underline{\boldsymbol{u}_{\mathrm{i}}} = \min(\mathrm{i}^{\mathrm{th}} \text{ row from } \mathbf{U}_{\mathrm{MC}}),$
  - assembly of $\underline{\boldsymbol{u}}$ from all $\underline{\boldsymbol{u}_{\mathrm{i}}}$,
- Finding the supremum:
  - for i-th node $\overline{\boldsymbol{u}_i} = \max(\mathrm{i}^{\mathrm{th}} \text{ row from } \mathbf{U}_{\mathrm{MC}}),$
  - assembly of $\overline{\boldsymbol{u}}$ from all $\overline{\boldsymbol{u}_i}$.

4th step: Calculation of inf-sup stresses in elements (resp. nodes) $[\underline{\sigma_{\mathrm{j}}}, \overline{\sigma_{\mathrm{j}}}]$ from the vector of displacements $[\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}]$.

2.1.2. Eigenvalue analysis:

1st step: generation of a random matrix (uniform distribution)

$\mathbf{X}_{\mathrm{MC}} = [\boldsymbol{x}_1, \ldots \boldsymbol{x}_{\mathrm{m}}], (\mathrm{m} \approx 5000 \div 100000),$

2nd step: Modal and spectral analysis:

- calculation of the i-th eigenvalue for j-th realization:
  $\lambda_{\mathrm{i\_j}} \rightarrow [\mathbf{K}(\boldsymbol{x}_{\mathrm{j}}) - \lambda_{\mathrm{i\_j}} \cdot \mathbf{M}(\boldsymbol{x}_{\mathrm{j}})] \cdot \boldsymbol{v}_{\mathrm{i\_j}} = \mathbf{0}$ for $\mathrm{j} = 1, \ldots, \mathrm{m},$
- assembly of the vector of solutions $\boldsymbol{\lambda}_{\mathrm{i}}$ for i-th eigenvalue

3rd step: determination of the inf-sup solution $[\underline{\lambda_{\mathrm{j}}}, \overline{\lambda_{\mathrm{j}}}]$:

- Finding the infimum of the *i*–th eigenvalue: $\underline{\lambda_{\mathrm{i}}} = \min(\boldsymbol{\lambda}_{\mathrm{i}}),$
- Finding the supremum of the *i*–th eigenvalue: $\overline{\lambda_{\mathrm{i}}} = \min(\boldsymbol{\lambda}_{\mathrm{i}}).$

2.2. Algorithm COM1

The COM1 algorithm is based on search of inf-sup solution for strictly boundary values of all uncertain parameters. However this algorithm has a disadvantage, it will not find a solution inside the acceptable region, but only in four "globally obtained" boundary points. Accuracy is limited to co called "proportional" uncertainty of parameters.

2.2.1. Static analysis

1st step: Calculation of inf-sup displacements $[\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}]$:

- calculation of the infimum: $\underline{\boldsymbol{u}} = \min\left\{\underline{\mathbf{K}}^{-1} \cdot \underline{\boldsymbol{f}}, \underline{\mathbf{K}}^{-1} \cdot \overline{\boldsymbol{f}}, \overline{\mathbf{K}}^{-1} \cdot \underline{\boldsymbol{f}}, \overline{\mathbf{K}}^{-1} \cdot \overline{\boldsymbol{f}}\right\},$
- calculation of the supremum: $\overline{\boldsymbol{u}} = \max\left\{\underline{\mathbf{K}}^{-1} \cdot \underline{\boldsymbol{f}}, \underline{\mathbf{K}}^{-1} \cdot \overline{\boldsymbol{f}}, \overline{\mathbf{K}}^{-1} \cdot \underline{\boldsymbol{f}}, \overline{\mathbf{K}}^{-1} \cdot \overline{\boldsymbol{f}}\right\}.$

2nd step: calculation of inf-sup stresses in elements (resp. nodes) $\left[\underline{\sigma_j}, \overline{\sigma_j}\right]$ from the vector of displacements $[\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}]$.

2.2.2. Eigenvalue analysis

- calculation of the infimum of the i-th eigenvalue:
  $\underline{\lambda_i} = \min\{[\underline{\mathbf{K}} - \lambda_i \cdot \underline{\mathbf{M}}] \cdot \boldsymbol{v}_i = \mathbf{0}, [\underline{\mathbf{K}} - \lambda_i \cdot \overline{\mathbf{M}}] \cdot \boldsymbol{v}_i = \mathbf{0}, [\overline{\mathbf{K}} - \lambda_i \cdot \underline{\mathbf{M}}] \cdot \boldsymbol{v}_i = \mathbf{0}, [\overline{\mathbf{K}} - \lambda_i \cdot \overline{\mathbf{M}}] \cdot \boldsymbol{v}_i = \mathbf{0}\}$,
- calculation of the supremum of the i-th eigenvalue:
  $\overline{\lambda_i} = \max\{[\underline{\mathbf{K}} - \lambda_i \cdot \underline{\mathbf{M}}] \cdot \boldsymbol{v}_i = \mathbf{0}, [\underline{\mathbf{K}} - \lambda_i \cdot \overline{\mathbf{M}}] \cdot \boldsymbol{v}_i = \mathbf{0}, [\overline{\mathbf{K}} - \lambda_i \cdot \underline{\mathbf{M}}] \cdot \boldsymbol{v}_i = \mathbf{0}, [\overline{\mathbf{K}} - \lambda_i \cdot \overline{\mathbf{M}}] \cdot \boldsymbol{v}_i = \mathbf{0}\}$.

2.3. Algorithm COM2

The COM2 algorithm is based on search of inf-sup solution for all combinations of boundary values of uncertain input parameters. A disadvantage of this algorithm is that the algorithm will not find an inf-sup solution inside the acceptable region, but only at boundary point, while accuracy is better than algorithm COM1.

2.3.1. Static analysis

1st step: calculation of the realization matrix $\mathbf{X_2}$, i.e. 2^n inf–sup combinations, (*n* – number of uncertain system parameters),

$\mathbf{X}_{\mathrm{COM2}} = [\boldsymbol{x}_1, \ldots \boldsymbol{x}_m], (m = 2^n)$,

2nd step: Calculation of the displacement matrix

$\mathbf{U}_{\mathrm{COM2}} = [\mathbf{K}(\boldsymbol{x}_1)^{-1} \cdot \boldsymbol{f}(\boldsymbol{x}_1), \ldots, \mathbf{K}(\boldsymbol{x}_m)^{-1} \cdot \boldsymbol{f}(\boldsymbol{x}_m)]$,

3rd step: Calculation of displacements $[\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}]$:

- Finding the infimum:
  - for i-th node $\underline{\boldsymbol{u}_i} = \min(\mathrm{i^{th}\ row\ from\ } \mathbf{U}_{\mathrm{COM2}})$,
  - assembly of $\underline{\boldsymbol{u}}$ from all $\underline{\boldsymbol{u}_i}$,
- Finding the supremum:
  - for i-th node $\overline{\boldsymbol{u}_i} = \max(\mathrm{i^{th}\ row\ from\ } \mathbf{U}_{\mathrm{COM2}})$,
  - assembly of $\overline{\boldsymbol{u}}$ from all $\overline{\boldsymbol{u}_i}$.

4th step: Calculation of stresses in elements (resp. nodes) $\left[\underline{\sigma_j}, \overline{\sigma_j}\right]$ from vector of displacements $[\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}]$.

2.3.2. Eigenvalue analysis

1st step: Calculation of the realization matrix $\mathbf{X}_2$, i.e. 2^n inf–sup combinations, (*n* – number of uncertain system parameters),

$\mathbf{X}_{\mathrm{COM2}} = [\boldsymbol{x}_1, \ldots \boldsymbol{x}_m], (m = 2^n)$,

2nd step: Modal and spectral analysis:

- calculation of the i-th eigenvalue for the j-th realization:
  $\lambda_{i\_j} \rightarrow [\mathbf{K}(\boldsymbol{x}_j) - \lambda_{i\_j} \cdot \mathbf{M}(\boldsymbol{x}_j)] \cdot \boldsymbol{v}_{i\_j} = \mathbf{0}$ for j = 1, …, m,
- assembly of the vector of solutions $\boldsymbol{\lambda}_i$ for the i-th eigenvalue

3rd step: determination of inf-sup solution $\left[\underline{\lambda_j}, \overline{\lambda_j}\right]$:

- finding the infimum of the i-th eigenvalue: $\underline{\lambda_i} = \min(\boldsymbol{\lambda}_i)$,
- finding the supremum of the i-th eigenvalue: $\overline{\lambda_i} = \max(\boldsymbol{\lambda}_i)$.

2.4. Algorithm OPT

The OPT algorithm is based on search of inf-sup solution by use of chosen optimization algorithm. It is a procedure similar to MC, but more effective in terms of analyses usage (many fewer). Advantage of this method is that it will find inf-sup solution, which is located inside the acceptable region and not only in boundary points. The searching process is realized by multiple optimization methods, depending on the type of the solved problem. Standardly is for the optimization used genetic algorithm as a powerful tool for global optimization, or Nelder-Mead simplex algorithm [2]. A serious problem is the formulation of objective function during search of infimum, resp. supremum. It often depends on the experience of the programmer.

2.4.1. Static analysis

1st step: Calculation of inf-sup displacements $[\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}]$:

- finding of the infimum:
  - for the i-th node we are searching for $\min\left\{\underline{u_i}(\underline{\boldsymbol{x}}_{\mathrm{opt}}) \rightarrow \left[\mathbf{K}(\underline{\boldsymbol{x}}_{\mathrm{opt}})^{-1} \cdot \boldsymbol{f}(\underline{\boldsymbol{x}}_{\mathrm{opt}})\right]\right\}$
  - assembly of $\underline{\boldsymbol{u}}$ from all $\underline{\boldsymbol{u}}_{\mathrm{i}}$,
- finding of the supremum:
  - for the i-th node we are searching for $\max\left\{\overline{u_{\mathrm{i}}}(\overline{\boldsymbol{x}}_{\mathrm{opt}}) \rightarrow \left[\mathbf{K}(\overline{\boldsymbol{x}}_{\mathrm{opt}})^{-1} \cdot \boldsymbol{f}(\overline{\boldsymbol{x}}_{\mathrm{opt}})\right]\right\}$
  - assembly of $\overline{\boldsymbol{u}}$ from all $\overline{u_{\mathrm{i}}}$.

2nd step: Calculation of stresses in elements (resp. nodes) $\left[\underline{\sigma_{\mathrm{j}}}, \overline{\sigma_{\mathrm{j}}}\right]$ from the vector of displacements $[\underline{\boldsymbol{u}}, \overline{\boldsymbol{u}}]$.

2.4.2. Eigenvalue analysis

- calculation of the infimum of the i-th eigenvalue:
  $\underline{\lambda_{\mathrm{i}}}(\underline{\boldsymbol{x}}_{\mathrm{opt}}) \rightarrow$ find $\underline{\boldsymbol{x}}_{\mathrm{opt}}$ for min. λ_{i}, i.e. $\underline{\lambda_{\mathrm{i}}} \rightarrow \left[\mathbf{K}(\underline{\boldsymbol{x}}_{\mathrm{opt}}) - \underline{\lambda_{\mathrm{i}}} \cdot \mathbf{M}(\underline{\boldsymbol{x}}_{\mathrm{opt}})\right] \cdot \underline{\boldsymbol{v}_{\mathrm{i}}} = 0$,
- calculation of the supremum of the i-th eigenvalue:
  $\overline{\lambda_{\mathrm{i}}}(\overline{\boldsymbol{x}}_{\mathrm{opt}}) \rightarrow$ find $\overline{\boldsymbol{x}}_{\mathrm{opt}}$ for max. λ_{i}, i.e. $\overline{\lambda_{\mathrm{i}}} \rightarrow \left[\mathbf{K}(\overline{\boldsymbol{x}}_{\mathrm{opt}}) - \overline{\lambda_{\mathrm{i}}} \cdot \mathbf{M}(\overline{\boldsymbol{x}}_{\mathrm{opt}})\right] \cdot \overline{\boldsymbol{v}_{\mathrm{i}}} = 0$.

Regarding finite element analyses it has to be noted, that based on the experience of the author, the use of mentioned numeric procedures is appropriate particularly for algorithms COM2 and OPT.

3. Application and comparison - solution of truss structure with interval parameters

Considering different uncertain parameters the numerical interval stress-strain study of a three-dimensional truss structure was performed [11, 12, 13]. The geometry of the structure is presented on Fig. 1. The truss structure was loaded by forces F in all upper nodes of the structure. The truss structure consists of 70 nodes and 257 bars.

The certain and uncertain model parameters are defined as follows:

- element mass density $\rho = 7800\ \mathrm{kg \cdot m^{-3}}$,
- Young‘s modulus $\mathrm{E} = 2.1 \cdot 10^{11}\mathrm{Pa}$,
- cross-section areas $\mathrm{A} = 0.0015\ \mathrm{m}^2$,
- force $\mathrm{F} = 1000\ \mathrm{N}$.

The force, cross-section area and Young’s modulus were used as the uncertain parameters and mathematically modeled as follows:

$$\langle \mathrm{F} \rangle = \mathrm{F} \cdot (1+\gamma) \quad , \quad \langle \mathrm{A} \rangle = \mathrm{A} \cdot (1+\gamma) \quad and \quad \langle \mathrm{E} \rangle = \mathrm{E} \cdot (1+\gamma).$$

The uncertainty degree was considered 50%, i.e. γ=0,5.

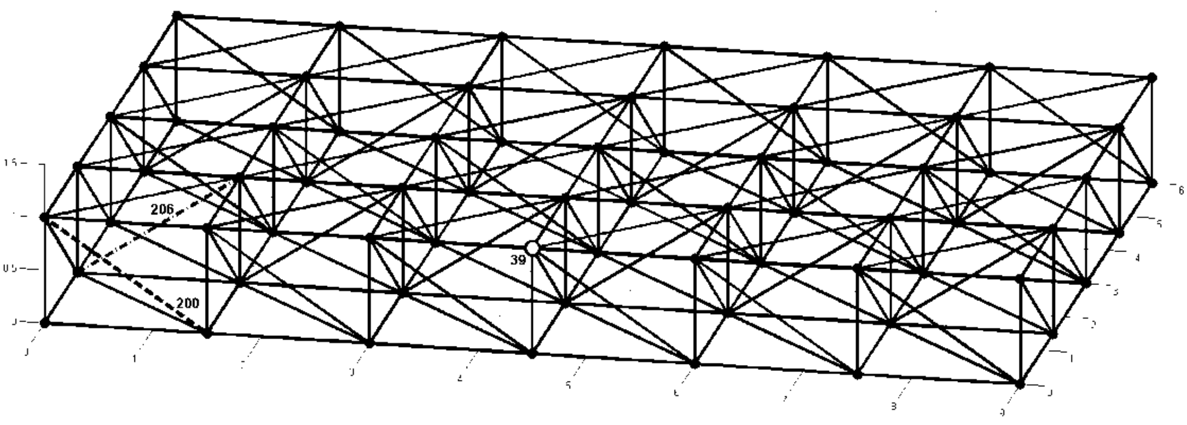

Fig. 1 FE model of analyzed truss structure, (dimensions in [m]) and identification of the maximal (bar No. 200), minimal (bar No. 206) stress values and maximal displacement (node No. 39)

The purpose of this study will be to compare the efficiency and exactness of the proposed methods MC, COM1, COM2 and OPT. The results of the MC analysis will be considered as reference values and will be used for the construction of the solution map. In the case of MC method, 10000 random inputs were generated; they were evaluated and properly processed to inf-sup solutions.

The maximal and minimal inf-sup stress values are summarized in Tab. 2 and maximal displacement shows Tab. 3.

Tab. 2 Stress inf/sup results for the chosen bars [MPa]

Bar No.	Stress [MPa]			
	COM1	COM2	OPT	MMC
200	<2,44 ; 3,11>	<2,30 ; 3,31>	<2,30 ; 3,31>	<2,54 ; 3,04>
206	<-4,83 ; -3,79>	<-5,13 ; -3,57>	<-5,13 ; -3,57>	<-4,72 ; -3,95>

Tab. 3 Displacement inf/sup result for the chosen node [mm]

Node No.	Displacement [mm]			
	COM1	COM2	OPT	MMC
39	<0, 29 ; 0, 30>	<0, 22 ; 0, 39>	<0, 22 ; 0, 32>	<0, 13 ; 0, 17>

Mapping of the generated input data by MC method for bar with min stress and for node with max displacement is shown on Figs. 2–3. Stress solution on the bar No. 206 for various uncertainties is shown on Fig. 4. Displacement solution on the node No. 39 for various uncertainties is shown on Fig. 5.

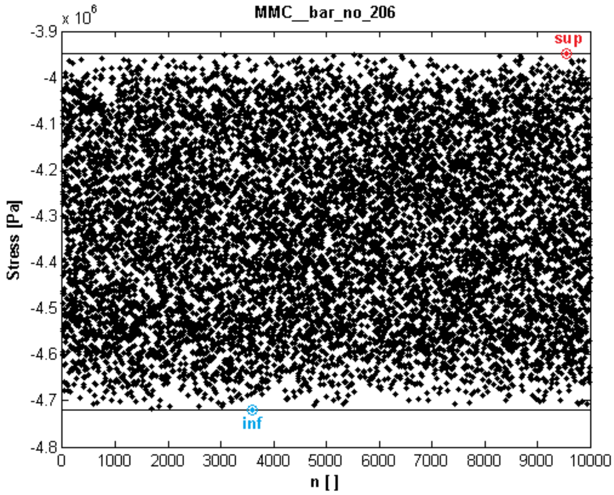

Fig. 2 Mapping of the generated input data for bar No. 206

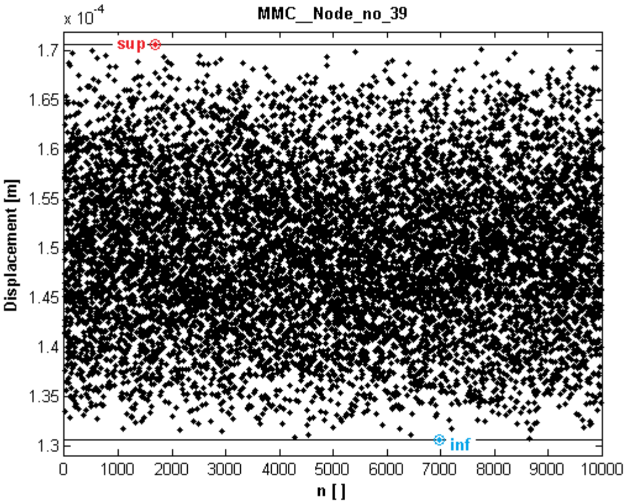

Fig. 3 Mapping of the generated input data for node No. 39

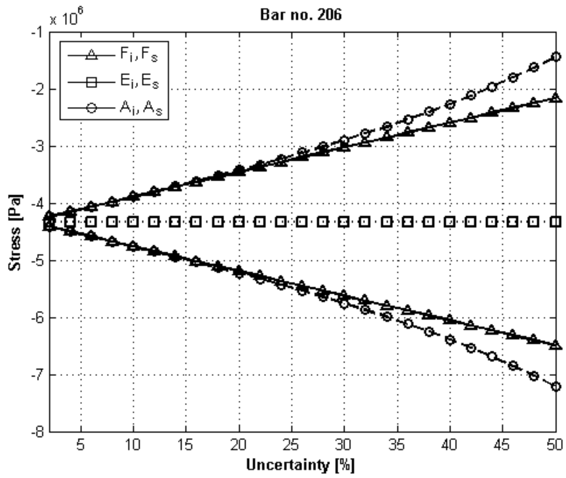

Fig. 4 Stress solution of bar No. 206 (max uncertainty 50%)

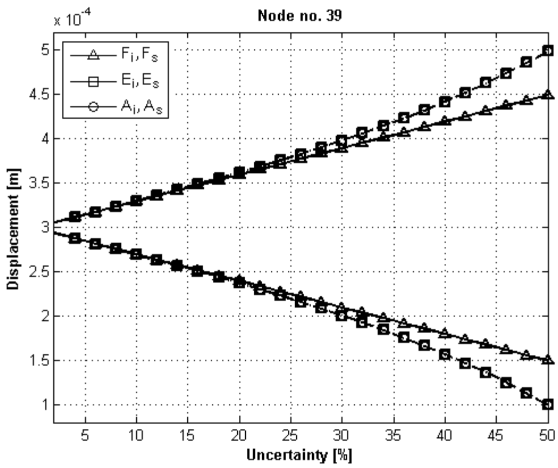

Fig. 5 Displacement solution on node No. 39 (max uncertainty 50%)

4. Conclusion

Our paper presents computational algorithms and their applications in an interval structural analysis. The use of the interval arithmetic provides a new possibility of quality and reliability evaluation of analyzed objects. It shows the stress-strain solution efficiency for solving problems including uncertain parameters with various interval width.

The analyses results can be summarized as follows:

- COM2 method provides decent results, but it is limited due to the exponential growth of the analyses number for complicated problems,
- OPT method provides good results and is suitable for complicated problems because it does not need so many analyses as in the cases of the MC or COM2 methods,
- the cross-section area as uncertain parameter has the most significant influence on stress solution,
- the cross-section area and Young's modulus as uncertain parameters have the most significant influence on displacement solution.

Acknowledgement

The work has been supported by the grant projects VEGA No.1/0234/13 and KEGA No.004ŽU-4/2012.

References

[1] R.E. Moore: Interval Analysis. Prentice Hall, Englewood Cliffs : New Jersey, 1966.
[2] Z. Kulpa, A. Pownuk, I. Skalna: Analysis of linear mechanical structures with uncertainties by means of interval methods, Computer Assisted Mechanics and Engineering Sciences, 1998, Vol. 5, Poland, pp. 443–477.
[3] I. Elishakoff, D. Duan: Application of Mathematical Theory of Interval Analysis to Uncertain Vibrations, Proc. of NOISE-CON'94, Ft. Lauderdale, Florida, pp. 519-524, 1994.
[4] R.L. Muhanna, R.L. Mullen: Sharp enclosure for material uncertainty in solid and structural mechanics – interval based approach, 8th ASCE Spec. Conference on Probabilistic Mechanics and Structural Reliability, 2000, electronic text: http://ecivwww.cwru.edu/civil/rlm/P331.pdf
[5] K.J. Bathe, E.L. Wilson: Numerical Methods in Finite Element Analysis. New Jersey : Prentice–Hall, 1976. 528 p. ISBN 0–13–627190–1.
[6] V. Dekýš, A. Sapietová, R. Kocúr: On the reliability estimation of the conveyer mechanism using the Monte Carlo method. Machine Dynamics Problems, 2006, Vol. 30, No 3, pp. 58-64, ISSN 0239-7730.
[7] A. Neumaier: Interval Methods for Systems of Equations. Cambridge University Press : Cambridge, 1990.
[8] H. Zhang: Nondeterministic Linear Static Finite Element Analysis: An Interval Approach. School of Civil and Environmental Engineering : Georgia Institute of Technology, 2005.
[9] M. Sága, M. Vaško: Algorithmization of Interval Structural Analysis. In: J. of Engineering, Annals of Faculty of Engineering Hunedoara, Tome VI, 2008, ISSN 1584-2665, pp. 149–160.
[10] M. Vaško: Application of Genetic Algorithms for Solving of Mechanical Systems with Uncertain Parameters. PhD. thesis, Žilina, 2007.
[11] S.H. Chen, X.W. Yang: Interval Finite Element Method for Beam Structures. In: Finite Elem. Anal. Des., 34, pp. 75-88, 2000.
[12] P. Kopas, R. Melicher, M. Handrik: FE analysis of thermo-plastic material model using for bearing steels. In: Applied mechanics 2008, 10th International Scientific Conference : April 7-10, 2008, Wisła, Poland, 2008. ISBN 978-83-60102-49-7.
[13] L. Jakubovičová, P. Kopas: Contribution to Stress and Residual Strain Analyse of the Welded Specimen, Transactions of the University of Košice, No. 3, Research reports from the universities of Košice, 2012, p. 57-64, ISSN 1335-2334.